

NASA Perspective on Requirements for Development of Advanced Methods Predicting Unsteady Aerodynamics and Aeroelasticity

David M. Schuster
Technical Fellow for Aerosciences
NASA Engineering and Safety Center
Hampton, VA

Outline:

I. Abstract

Over the past three years, the National Aeronautics and Space Administration (NASA) has initiated design, development, and testing of a new human-rated space exploration system under the Constellation Program. Initial designs within the Constellation Program are scheduled to replace the present Space Shuttle, which is slated for retirement within the next three years. The development of vehicles for the Constellation system has encountered several unsteady aerodynamics challenges that have bearing on more traditional unsteady aerodynamic and aeroelastic analysis. This paper focuses on the synergy between the present NASA challenges and the ongoing challenges that have historically been the subject of research and method development. There are specific similarities in the flows required to be analyzed for the space exploration problems and those required for some of the more nonlinear unsteady aerodynamic and aeroelastic problems encountered on aircraft. The aggressive schedule, significant technical challenge, and high-priority status of the exploration system development is forcing engineers to implement existing tools and techniques in a design and application environment that is significantly stretching the capability of their methods. While these methods afford the users with the ability to rapidly turn around designs and analyses, their aggressive implementation comes at a price. The relative immaturity of the techniques for specific flow problems and the inexperience with their broad application to them, particularly on manned spacecraft flight system, has resulted in the implementation of an extensive wind tunnel and flight test program to reduce uncertainty and improve the experience base in the application of these methods. This provides a unique opportunity for unsteady aerodynamics and aeroelastic method developers to test and evaluate new analysis techniques on problems with high potential for acquisition of test and even flight data against which they can be evaluated. However, researchers may be required to alter the geometries typically used in their analyses, the types of flows analyzed, and even the techniques by which computational tools are verified and validated. This paper discusses these issues and provides some perspective on the potential for new and innovative approaches to the development of methods to attack problems in nonlinear unsteady aerodynamics.

II. Introduction

In November 2005, the National Aeronautics and Space Administration (NASA) published their Exploration System Architecture Study¹ initiating design and development of a new human-rated space exploration system to replace the aging Shuttle Transportation System (STS) and carry humans to the moon and beyond. This new

system, developed under the NASA Constellation (Cx) Program, involves development of a vehicle much akin to that used in the highly successful Apollo Program². The new vehicle designated Orion/Ares, Figure 1, uses a capsule system for the Orion Crew Exploration Vehicle (CEV) which is very similar in Outer Mold Line (OML) design to the former Apollo capsule of the 1960's. The new launch vehicle, designated Ares, uses a STS-heritage, recoverable/reusable solid rocket motor as its first-stage booster and a liquid-fueled second stage to transport the CEV to earth orbits where it can rendezvous with the International Space Station (ISS) or a separately launched earth departure vehicle to explore beyond earth orbit. Initial Orion/Ares designs are expected to be operational within the next decade, and extensive analysis and testing of designs is under way. Preliminary flight tests of the vehicle are expected to begin late in 2008 and steadily continue through the first manned flight of the vehicle. The aggressive schedule and high priority of this development have forced engineers to rely heavily on analytical and computational methods to perform conceptual and preliminary design of the vehicle³. NASA is responsible for the development and delivery of the aerodynamic and aerothermodynamic databases for both the Orion and Ares components. This is being accomplished through a tightly coordinated effort involving the use of theoretical and experimental techniques. In this effort, high-order Computational Fluid Dynamics (CFD) methods are being utilized in a principal design and analysis mode to a much higher degree than for any project in NASA history. CFD is being used to evaluate and select designs, often eliminating designs without any wind tunnel testing. Wind tunnel experiments are being used to verify computed performance, and reduce uncertainties for particularly difficult and extreme flow cases and geometries for which a high degree of confidence in the computational methods is not evident. This approach has led to the use of CFD techniques for problems where little previous experience is available. Among these situations is the case of unsteady, separated flows across the Mach number range from subsonic to hypersonic flight.

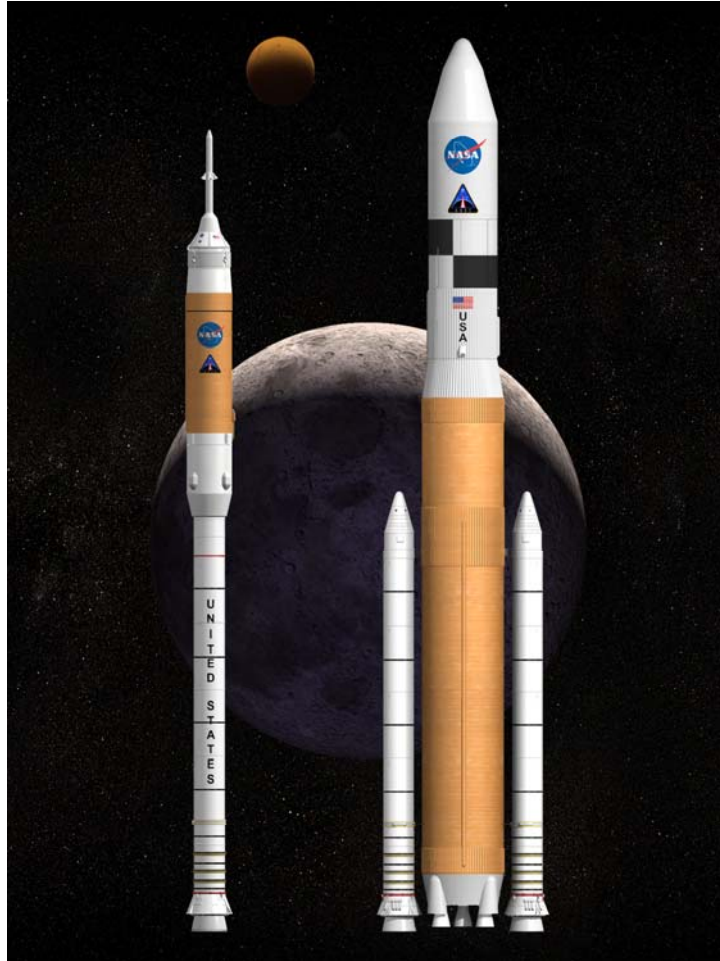


Figure 1. NASA's Orion/Ares spacecraft

There are two primary configurations of the Orion CEV that must be aerodynamically evaluated. There is the basic Crew Module (CM), shown in Figure 2, in which the crew resides for most of any particular mission. From an aerodynamic analysis standpoint the primary areas of interest occur during ascent as the vehicle traverses the entire Mach range from low subsonic to high supersonic flight, and during orbit reentry where the vehicle sees a similar range of operating conditions, adding hypersonic flight, but in a much different vehicle orientation than during ascent, namely heat-shield forward.

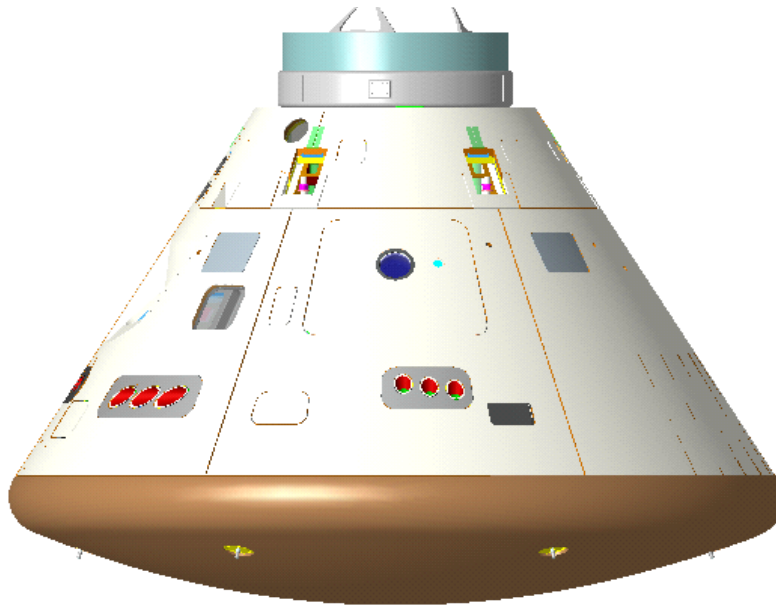


Figure 2 The Orion Crew Exploration Vehicle

The second CEV configuration involves operation in the case of a launch abort. In this configuration, the CM separates from the launch vehicle through the use of an escape tower system. The Launch Abort Vehicle (LAV), Figure 3, includes the CM and a large tower that includes a number of solid rocket motors to initiate and control the abort. In the event of an abort, the vehicle flies in a wide range of orientations from zero angle-of-attack (AOA) with the tower forward to 180 degrees AOA with the exposed heatshield forward. Aborts employing the LAV can be initiated at any time prior to or during the operation of the first stage and up to 30 seconds into the operation of the second stage booster. Therefore the launch vehicle operates over an extreme range of flight conditions ranging from subsonic to high supersonic flight and altitudes from sea level to in excess of 100,000 ft. In addition, the LAV must maneuver near the mid-point of any abort scenario orienting the heatshield of the vehicle forward to facilitate the deployment of the parachute recovery system. This requires the vehicle to perform a flip maneuver that changes the angle-of-attack from near-zero AOA to near-180 degrees AOA. Further complicating the aerodynamic analysis of this configuration is the complex jet interactions as the various abort and control motors fire to initiate and control the LAV flight⁴.

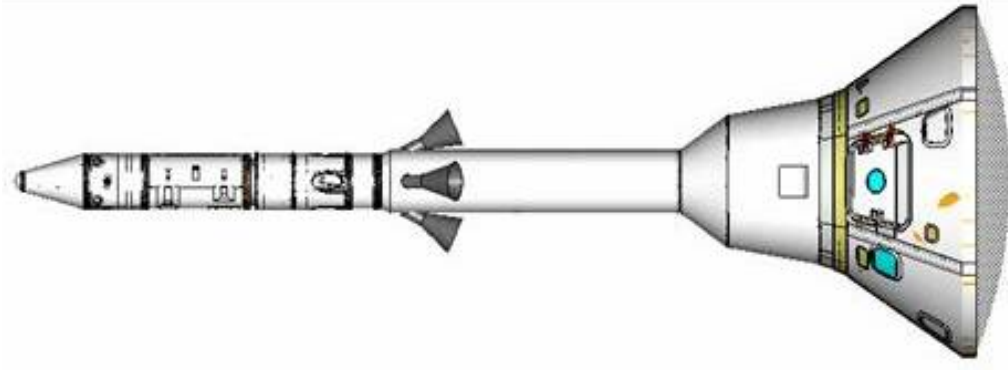


Figure 3. The Orion Launch Abort Vehicle

The Ares I Crew Launch Vehicle (CLV) is a tall, slender rocket that has a significant diameter change between the first and second stage boosters. It also employs a number of large protuberances in the form of ullage, separation, and reaction control motors that control the vehicle ascent. The interface between the first and second stage is a point of reduced structural stiffness for the vehicle, and due to its long length, the bending incurred at this joint due to ascent loads results in significant bending deformation of the stack that can influence the control and aerodynamic performance of the CLV. The vehicle must traverse the atmosphere at speeds ranging from subsonic to high supersonic with the maximum dynamic pressure and correspondingly maximum aerodynamic loads on the vehicle occurring in the transonic flight regime.

Thus, the Ares and Orion vehicles operate over a range of atmospheric flight conditions that span the complete speed range from subsonic to hypersonic, angles-of-attack from zero to 180 degrees, and an extremely broad range of Reynolds numbers. Rarely is it required that a single vehicle be designed over such a wide range of flight conditions. These conditions challenge the full breadth of our computational, experimental, and flight test capabilities. The fact that this is a human-rated system along with the aggressive schedule that the agency has laid out for the development of this vehicle provides the motivation and the means for extensive ground and flight testing to verify designs. Therefore, this design and development effort provides a unique opportunity for researchers and developers to test, verify, and validate their methods, albeit on geometries that may be unfamiliar to more traditional aerodynamic method development. The geometric characteristics of the vehicle include bluff bodies and large angular changes that further complicate the aerodynamic analysis. However, these characteristics provide additional opportunities for developers of unsteady aerodynamic methods to move the state-of-the-art forward with unprecedented test and flight data against which to compare their methods.

III. Orion and Ares Aerodynamic Features Requiring Analysis and Evaluation.

During ascent the CEV sees flows that must negotiate sharp, often large-angle, corners, resulting in strong shocks in the transonic flight regime and separated flows across the flight regime, as depicted in Figure 4. The fluctuating pressure fields generated by these phenomena comprise the aeroacoustic environment in which the vehicle must operate,

and aeroacoustic performance has become a driving design factor for the vehicle during nominal ascent. During reentry, the CEV orients itself with the heatshield forward to enter the atmosphere, and as such presents a large bluff body to the flow. As the flow negotiates this body, it can separate on the lee side of the capsule, Figure 4, which if highly dynamic, can have significant impact on the flight stability of the vehicle as the capsule tries to maintain orientation for parachute deployment and flight under the parachute system.

Flowfield features of the LAV in forward flight are shown in Figure 5. In the tower-forward orientation, the LAV involves extensive bluff body flow as the entire heatshield is exposed to the flow and is operating as the trailing surface of the vehicle. The flow separates near the maximum-breadth shoulder of the heatshield and there is potential for extensive unsteady flow in this region. With the interaction of high temperature, reacting plumes from the launch abort and control motors, it is readily evident that this is an extremely challenging, dynamic, nonlinear aerodynamic problem.

The Ares launch vehicle while outwardly appearing to be aerodynamically simpler as a high fineness ratio body, also encounters flow situations that challenge today's prediction methodology. Areas of separated flow are encountered on the vehicle as flow negotiates the relatively large diameter changes and protuberances on the vehicle OML. These separations lead to unsteady aerodynamic environments that impact the performance of the vehicle as well as its overall structural and system design.

Orion Ascent

Orion Reentry

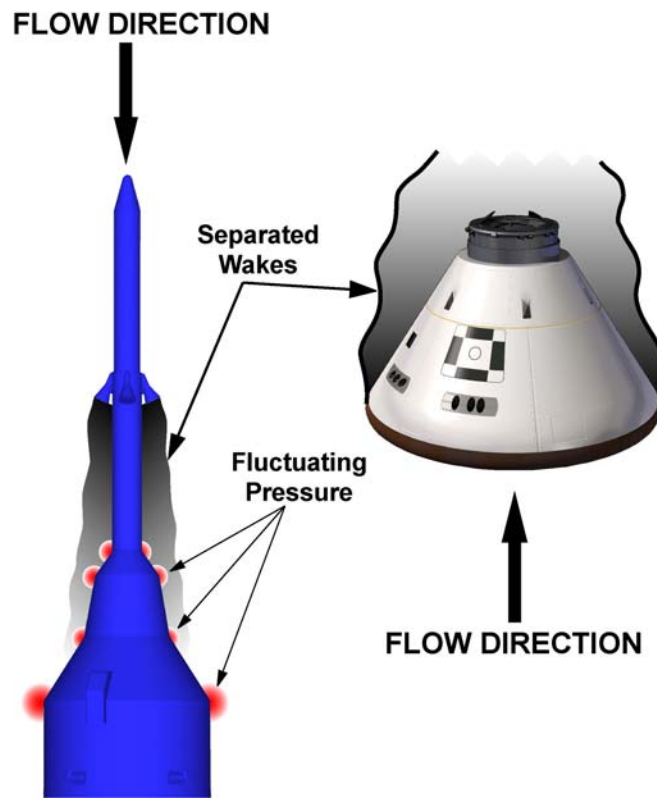


Figure 4. Orion ascent and reentry unsteady flow features

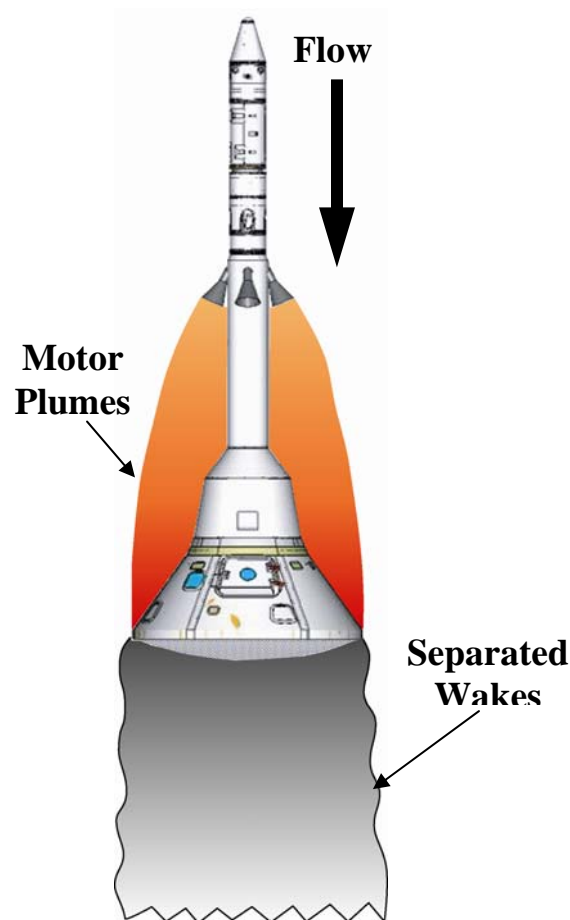


Figure 5. Orion LAV unsteady flow features.

Unsteady pressure environments generated by the Orion/Ares vehicle as it accelerates through the earth's atmosphere have a direct impact on the buffet and aeroacoustic loads applied to the vehicle structure, flight systems, and internal payloads, including humans. The methods by which these vehicles are qualified for human-rated flight, particularly for aeroacoustic loads, is placing added emphasis on the development of physics-based prediction methods capable of accurately simulating the unsteady pressure fields associated with flow nonlinearities generated by these vehicles.

The requirements and processes for qualification of these vehicles differ from aircraft. Due to the single-use character of some of the launch vehicle components, flight tests are extremely expensive and are typically limited to single-flight demonstrations of the critical vehicle components. In contrast, aircraft are usually flight tested and qualified over a range of flight parameters that clear the vehicle for flight anywhere within its design envelope. Due to the very limited availability of operational flight testing for launch vehicles and spacecraft, their systems and components are qualified through ground testing. To ensure that these tests cover the launch conditions plus potential dispersions, the ground tests are conducted with margins well beyond conditions expected during a nominal flight. In the case of the Orion/Ares spacecraft, the predicted unsteady pressure environments generating aeroacoustic loads are forcing engineers to

qualify the spacecraft flight article to conditions that are beyond the capability of current test facilities.

Thus the agency is faced with expensive upgrades of test facilities or reduction of the flight environments to levels at which the vehicle flight hardware can be effectively qualified in existing facilities. Consequently engineers are very interested in reducing the conservatism built into their prediction methods. In the case of unsteady pressure environments for aeroacoustic loads, empirical methods enveloping existing wind tunnel and launch vehicle flight data are used to make this prediction. In many cases, this empirical enveloping approach adds conservatism to the predicted environments which may not be required for the vehicle under development. The availability of accurate, efficient physics-based unsteady pressure prediction methods could reduce this conservatism and ensure that the vehicle is only qualified to levels appropriate for the flight conditions and environments actually encountered by the vehicle. The conditions that set these qualification requirements often occur in the transonic flight regime of the ascent trajectory and the flow in this region involves strong shocks and separated flows that trigger the natural unsteadiness leading to the unsteady pressure environments.

IV. NASA's Approach to Aerodynamic Analysis of the Constellation Exploration Vehicle

To aerodynamically evaluate the Cx exploration vehicle, NASA has adopted an aggressive approach that utilizes Computational Fluid Dynamics as the principal analysis tool for the evaluation and selection of vehicle concepts, and in fact the final aerodynamic analysis of the Cx vehicle. NASA is leveraging their extensive experience in applying these techniques to the analysis of the Space Shuttle, Figure 6. For the past 15 years, and particularly since the loss of the space shuttle Columbia in 2003, NASA has accelerated their implementation of CFD for the prediction of the varied, complex flowfields experienced by the shuttle during its ascent and reentry⁵. They are building off this experience to analyze the Cx exploration system utilizing an array of inviscid and viscous CFD methods. For inviscid analysis, Cx engineers are implementing the Cartesian grid Euler solver known as CART3D⁶ for cases where perfect gas assumptions are valid, and the FELISA⁷ code for reacting chemistry inviscid cases. For viscous flows, Cx is utilizing the OVERFLOW⁸ overset structured grid method and the USM3D⁹ unstructured grid method as their primary perfect gas analysis tools. They are also using the FUN3D¹⁰ unstructured grid method, which is in a more developmental state than the previous methods, but includes the capability to analyze non-perfect gases. USM3D is also undergoing modifications to include reacting gas chemistry, which will provide a redundant viscous reacting gas capability to the program. Thus, many of the methods being employed by the project have redundancy in the form of similar methods with differing formulations and different users exercising the methods. This redundancy is critical since the complexity of the flows being analyzed can result in large differences in results even with what would seemingly be small changes in input, geometry, or flowfield modeling. Having multiple analysis streams being exercised by independent users provides a consistency check for the analysis and also provides uncertainty and parameter sensitivity information to the project engineers.

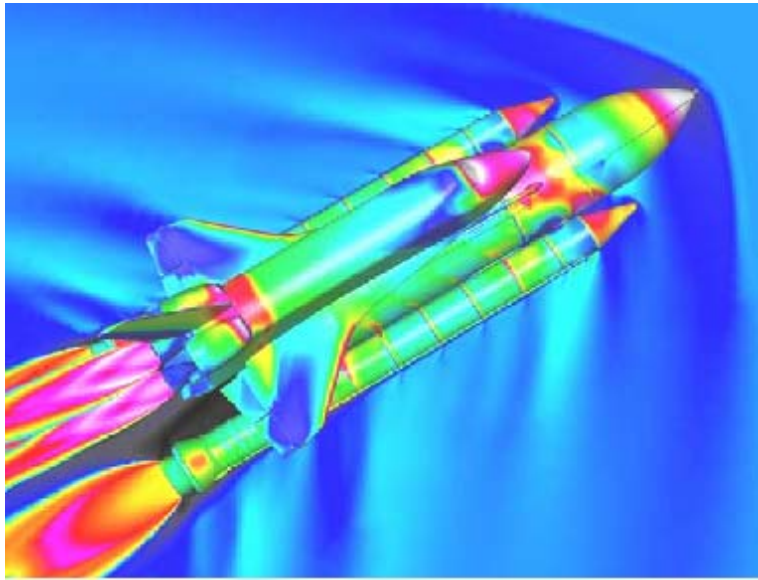


Figure 6. CFD computation of the NASA Space Shuttle stack configuration flowfield with propulsion simulation.

The above methods employed by the Cx Program have been developed, validated and applied to problems primarily in the steady flow regime. However, as demonstrated in the previous section, the Cx vehicle presents many opportunities for unsteady flow in its nominal flight envelope. As engineers have applied their CFD tools to the Cx configurations these unsteady instances have been encountered, even though the methods are being applied in a steady flow analysis mode. This has led some engineers to simply run the methods in a time-accurate mode from initiation rather than wasting an additional step to analyze the vehicle in a steady mode before being forced to run method in a time accurate mode. Another approach has been to run the methods in a steady state mode, then average the force and moment results over the last 1000 or more iterations to remove the possibility of unsteadiness biasing these data. This approach is effective to get mean forces and moments, and may also be effective for mean pressures, but it is ineffective and inaccurate when the amplitude, frequency, and phase of the unsteady flow are important to the aerodynamic analysis. There are a number of important unsteady flow characteristics on the Cx vehicle that require accurate prediction of the unsteady amplitude, frequency, and phase characteristics and require a true unsteady flow analysis.

Accurate prediction of unsteady flow has long been a priority concern for NASA researchers, particularly in the Aeroelasticity discipline specialty. Prediction of unsteady flow dates back to the 1930's¹¹ with the development of lifting line theory and has enjoyed steady progress through the 1970's with the development of doublet lattice theory¹². Beginning in the 1970's, the development of Computational Fluid Dynamics (CFD) methodology commenced and for the last thirty years, CFD has enjoyed intense development and application, primarily focusing on steady flow prediction. As steady CFD methods have matured and their application has become increasingly routine, research and development has migrated into the unsteady flow arena.

V. Contrast Between Traditional Unsteady Aerodynamic Predictions and the Orion/Ares Analysis Problem

Much of the classical unsteady aerodynamic analysis capability in existence today is based on thin geometry, small disturbance assumptions, which lead to a linear set of equations that can be readily and efficiently solved. These classical methods have very limited application when investigating low length-to-diameter ratio bodies like the Orion capsule or the LAV. Their applicability broadens slightly for the Ares launch vehicle, but in the transonic and maximum dynamic pressure flight regimes where the flow is highly nonlinear these methods have little or no utility.

Similarly, CFD methods applied to unsteady aerodynamic problems have focused on attached or only mildly separated flows. The steady Reynolds Averaged Navier-Stokes (RANS) methods in broad use today do not have many of the geometric or flow nonlinearity assumptions and limitations of the classical methods, but the experience base and in some cases, the development philosophy behind these methods is increasing the data uncertainty resulting from these techniques when they are applied to the Cx exploration vehicle. This is particularly true in the critical transonic flight regime. The majority of experience with these methods resides in the analysis and prediction of nonlinear steady flows. The experience base for these codes when applied to nonlinear unsteady flow problems is extremely limited.

While the codes have been sporadically applied to unsteady flows, including some nonlinear flows in the transonic flight regime, the simulation approach using these codes is much different than that required for analysis of the Cx exploration vehicle. Most of the experience with unsteady CFD analyses involves the flow about bodies with oscillating or otherwise unsteady boundary conditions. In this case, the motion of the body provides a continuous input of unsteady energy to the system of equations and the characteristics of the most interesting, near-field flow is strongly influenced by the motion of the body, not necessarily the inherent unsteady physics of the flow. In addition, this class of unsteady aerodynamic analysis problems serves to effectively mask the performance, particularly the dissipative characteristics, of the numerical algorithm used to solve the unsteady equations of motion. The amplitude and frequency characteristics of the unsteady flow near a moving body are strongly influenced, if not dominated by the unsteady motion of the body. It isn't until the flow several, to many, characteristic body lengths away from the moving body is carefully examined that one can effectively assess the numerical performance of the unsteady aerodynamics solver. In fact, for many unsteady aerodynamic problems, flow far from the vehicle is seldom of any practical significance to the aerodynamicist, since it is the loads and local flow around the body that are of greatest importance. Therefore, unless the algorithm under evaluation is particularly dissipative to the point that it can noticeably alter the near-field flow, there is a relatively high probability that the algorithm will be able to predict the near-field unsteady aerodynamics of an oscillating or otherwise moving body.

This forced oscillation unsteady assumption is not a good one for the conditions under which the Cx exploration vehicle operates. In this case, the flow unsteadiness is

produced by the natural physics of the flow, not necessarily by the motion of the vehicle. In most cases, there is no forced oscillation motion to generate or sustain the energy present in the unsteady flow. Therefore, the simulation is much more sensitive to the detailed characteristics of the numerical algorithm used to capture the unsteadiness. The dissipative nature of the algorithm has a more pronounced effect on the ability of the method to capture the proper amplitude, frequency, and phase character of the unsteady flow. Thus the aeroacoustic and buffet environments for the vehicle, as predicted by computational methods, result in a significant amount of analysis uncertainty. This uncertainty, along with the lack of experience in the application of the computational techniques in existence today, requires that aeroacoustic environments for the Cx vehicle be predicted using empirical methods and historical data.

In more traditional aerodynamic applications, unsteady flow predictions involving separation represent cases that are off-design for most vehicles. Aircraft designs are often optimized for cruise flight where separation is avoided due to its large performance penalties. Even for aircraft required to maneuver at extreme flight conditions, separated flow is minimized due to its detrimental impact on maneuvering performance. Thus, conditions involving unsteady, separated flow phenomena are isolated enough that they often can be treated as special cases and are typically quantified through wind tunnel experiments and flight tests. Analytical and numerical predictions are usually only applied to on-design flight situations involving more benign, steady and attached flows. Thus our computational and analytical methods have been developed, and in some instances optimized, for steady, attached flows.

In contrast, the Orion and Ares exploration vehicles are optimized for their space performance, not necessarily for aerodynamic performance. They however, must traverse the atmosphere both enroute to and returning from space and predicting the aerodynamic performance of these vehicles is still extremely important to their operation, even if it isn't a principal design consideration in their development. So unlike aircraft, spacecraft may have significant regions of separated and nonlinear flow under nominal operating conditions. To effectively and accurately predict and simulate the nominal atmospheric flight of the Cx exploration vehicles, engineers must predict nonlinear separated and often unsteady flowfields.

VI. Shortcomings of the Present Methodology

As noted in Sections IV and V, development of methods to predict unsteady flows has been a topic of research for quite some time, and even the use of CFD to perform these predictions has enjoyed some development, though certainly not to the extent of steady CFD methods. However, this development and application has not aggressively attacked two important sets of problems that are critical to many of the difficult, nonlinear aeroelastic problems facing the aerospace industry today. The first is the prediction of naturally unsteady flow. The previously discussed forced oscillation problems are the natural choice for verification and validation of unsteady flow methods since these flow problems result in bounded, periodic amplitude and frequency responses that can be used

to quantitatively evaluate the methods. However, many of the unsteady flows that occur on modern aerospace vehicles are not a response to periodic motion, but rather a natural unsteady response to flow nonlinearities.

This leads into the second problem set that has not been thoroughly investigated using CFD, that being separated and separation onset flows. These separated flows play into the previously discussed naturally unsteady flow problems since the nonlinearity of the separation often triggers unsteadiness in the flow. These flows are decidedly more difficult to accurately capture and predict than those generated by forced oscillation since the amplitudes and frequencies of the unsteadiness are generated by the flow physics rather than an artificial unsteadiness imposed upon the flow. For the separated and naturally unsteady flow cases, the detailed characteristics of the CFD method, at an algorithmic level, become important because the ability of the algorithm to accurately capture the convective and diffusive nature of the flow has a direct and significant impact on the unsteady flow prediction. For this reason, the author asserts that the present approach of adapting CFD methods developed to predict steady flow problems to unsteady flows may be flawed, particularly for naturally occurring unsteady flows.

Steady flow algorithms have been optimized over the last thirty years to converge to a steady state as rapidly and efficiently as possible without adversely affecting the mean flow prediction. Therefore, the most efficient steady flow algorithms quickly damp transient oscillations from perturbations, whether numerical or physical, to drive the flow to a steady state as quickly as possible. The dissipative nature of the steady flow algorithms are masked in the simulation of many forced oscillation problems since the flow simulation involves a continuous influx of perturbation by the artificial forced oscillation of the geometry. However, the unsteadiness generated by a separated flow, particularly an incipient separated flow or other flow nonlinearity, is highly dependent on the ability of the algorithm to accurately simulate convective and diffusive characteristics, not to damp these transient characteristics to ensure a quick convergence to steady state.

To date, the prediction of naturally occurring and separated flows has not enjoyed a great deal of attention from algorithm developers, primarily due to the difficulty of the problem itself, the challenge of effectively verifying and validating any resulting algorithms, and the relatively narrow set of problems requiring this capability. Despite the severe nature of the problems and resulting consequences that have been attributed to these types of flows such as the F-16 Limit Cycle Oscillation¹³ and the F-18 E/F Abrupt Wing Stall¹⁴, addressing these problems has typically been accomplished through wind tunnel and flight testing. However, those interested in addressing these types of flows have a new opportunity in NASA's development of the Cx exploration vehicle.

Accepting this challenge will not be easy for developers, and they will likely have to change their paradigm for method development, verification, and validation. First, the geometries analyzed are significantly different from those traditionally analyzed in unsteady aerodynamic method development. Small perturbation wings such as the F-5 wing¹⁵ or the AGARD 445.6 wing¹⁶, which have enjoyed considerable attention from the

unsteady aerodynamics community over the past 20 years, have no resemblance and virtually no relevance to analysis of Cx exploration vehicle flowfields. So developers accepting this challenge will first be required to rethink the types of geometries on which they test their methods.

Second, and probably more significant, are issues surrounding verification and validation of naturally occurring unsteady flow, particularly those due to separation and separation onset. Developing methods to attack these problems will certainly be a challenge, and may require that developers examine their methods on an algorithm level before they will be able to make significant progress in this area. However, even if these issues are addressed, or are believed to have been addressed, verifying and validating the methods will still be a formidable challenge. Method verification and validation for these types of problems will require developers to risk that their methods may not necessarily correlate well with available data the first time out and, in fact, it might be quite some time before they can demonstrate that their methods are performing adequately. Naturally unsteady, separated, and separation onset flows can be very sensitive to the boundary conditions under which they are formed. Uncertainty and test-to-test variation in experimental and flight data simulating these flows can be significant, as demonstrated in Figure 7 copied from Reference 17. In this figure, the Limit Cycle Oscillation (LCO) amplitude of a transport wing in separated transonic flow is mapped against dynamic pressure and Mach number. The labels on the curves represent observed acceleration levels in g's of the wing motion, which relates directly to the amplitude of the unsteady aerodynamics on the wing. The "B" designation on some of the numbers indicates an observed "bursting" phenomenon which is due to a transient separation, which is often a characteristic of separation onset that triggers sporadically into fully developed separation. The mapping with dynamic pressure and Mach number is very complex and the data can, at best, only be described in pockets of LCO with ranges of deformation amplitude. The bursting observation further complicates the unsteady flow situation observed in the experiment making it even more difficult to quantitatively compare computational predictions with the experimental data. Concise quantitative data for this phenomenon cannot be derived from this test. This figure illustrates that it will take time and effort to acquire sufficient data to bound natural unsteady flow uncertainties and understand the sensitivity of the data to boundary conditions and surrounding flowfield variations. Given the nature of the problem, only when these uncertainty bounds and sensitivities are understood for a specific set of problems can method developers ascertain if their techniques fall within those bounds. In this regard the problem is much different than the steady flow problem where stationary, low variation data is generally available and method correlation and evaluation is relatively straightforward.

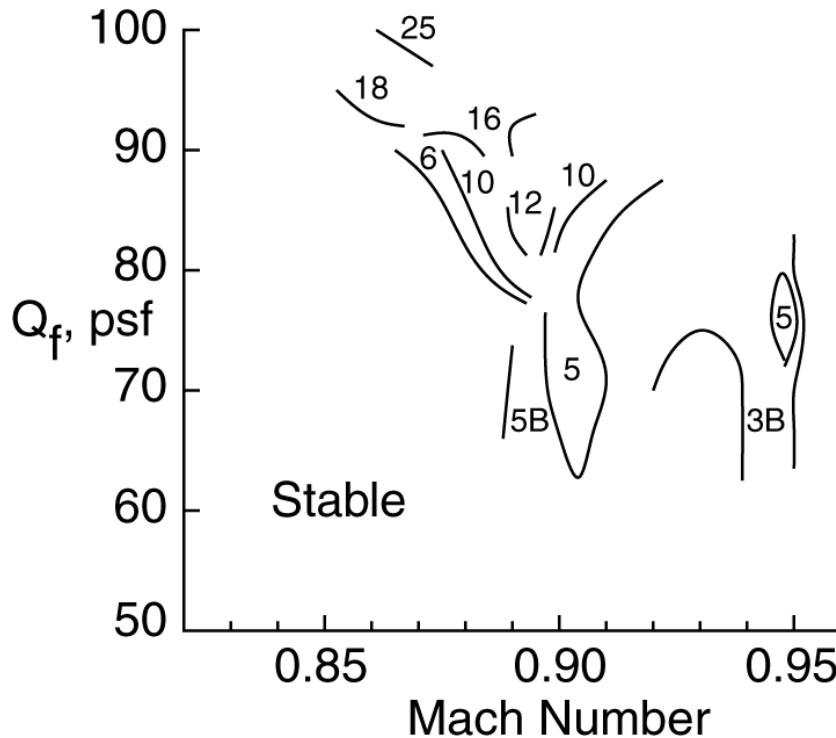


Figure 7. MAVRIC transport wing LCO amplitude as a function of dynamic pressure and Mach number, cf. Reference 16.

VII. Conclusion

The simulation of naturally unsteady, separated, and separation onset flows is not a new one for the aerospace community, and problems involving these types of flows have plagued engineers for decades. These flows are often difficult to capture experimentally and in flight test, and tests involving these types of flows are characterized by a high degree of data uncertainty and variation. These issues make development of methods to simulate these flows more difficult because it is hard to verify if one's method is adequately simulating the flow of interest. Beyond this issue is that of simply the difficulty in accurately capturing and simulating these flowfields. Many of the issues that relate directly to the accurate simulation of steady flows, such as turbulence modeling and sufficient grid refinement and geometric modeling will be important to simulation of naturally unsteady, separated, and separation onset flows. But the research community may also have to step back and examine their methods on an algorithmic level to ensure that the dissipative qualities of the methods that make them so robustly converge to a steady state for steady flows aren't inadvertently quenching the unsteadiness in naturally occurring unsteady flows. Steady CFD has enjoyed over thirty years of concentrated development and these methods are being increasingly relied upon to design and analyze our flight vehicles, minimizing expensive ground and flight testing in the process. The aerodynamic methods development community should not assume that these methods are

directly transferable to the simulation of naturally unsteady flows, and the algorithms and techniques used in these methods should be carefully investigated and, if necessary, reformulated for the unsteady flow problem.

The design and development of the Orion and Ares manned space exploration vehicles may provide a basis on which methods simulating naturally unsteady flows can be validated. The Constellation vehicle aspires to take man beyond low earth orbit for the first time in over 30 years. Though it is geometrically similar to the Apollo vehicle of the 1960's, its systems, performance and most notably its design, are drastically different. CFD is now the backbone of the aerodynamic and aerothermodynamic database development for the vehicle. While engineers are confident that the CFD can effectively simulate the majority of aerodynamic issues on the vehicle, there are some, such as aeroacoustic and buffet environments, which cannot be effectively simulated using this technique and engineers are forced to rely on ground and flight experiments and existing empirical data to perform their design trades. Thus there is potential for a significant unsteady aerodynamic database for naturally unsteady, separated, and separation onset flows. These data may be of value to the development community if they are willing to change the geometries that have been traditionally analyzed from the aerodynamically streamlined small disturbance airfoils and wings to the low aspect ratio bluff bodies of the Cx exploration vehicle. Developers will also have to risk that verification and validation of their methods may be as difficult a problem as developing the methods themselves. Hopefully the aggressive design philosophy, high priority, and importance of the Constellation mission will provide the motivation for developers to accept this challenge and move the state-of-the-art in unsteady aerodynamic method development forward.

VIII. References

1. "NASA's Exploration System Architecture Study," NASA-TM-2005-214062, National Aeronautics and Space Administration, November, 2005.
2. Seamans, R. C. Jr., "PROJECT APOLLO The Tough Decisions," Monographs in Aerospace History, Number 37, NASA-SP-2005-4537, National Aeronautics and Space Administration, 2005.
3. Greathouse, et al., "Crew Exploration Vehicle (CEV) Crew Module Shape Selection Analysis and CEV Aeroscience Project Overview," AIAA-2007-603, American Institute of Aeronautics and Astronautics, January, 2007.
4. Aftosmis, M and Rogers, S, "Effects of Jet-Interaction on Pitch-Control of a Launch Abort Vehicle," AIAA Paper AIAA-2008-1281, January 2008.
5. Gomez, R., et al., "STS-107 Investigation Ascent CFD Support," AIAA Paper AIAA-2004-2226, June 2004.
6. Aftosmis, M.J., Berger, M.J., and Adomavicius, G. D. "A Parallel Multilevel Method for Adaptively Refined Cartesian Grids with Embedded Boundaries," AIAA Paper AIAA-2000-0808, January 2000.
7. Bibb, K. L., Peraire, J., and Riley, C. J., "Hypersonic Flow Computations on Unstructured Meshes," AIAA Paper 97-0625, January 1997.
8. Buning, P. G., et al., "Numerical Simulation of the Integrated Space Shuttle Vehicle in Ascent," AIAA Paper AIAA-88-4359, August 1988.

9. Frink, N. T., "Tetrahedral Unstructured Navier-Stokes Method for Turbulent Flows," *AIAA Journal*, Vol. 36, No. 11, Nov. 1998, pp. 1975–1982
10. Anderson, W. K. and Bonhaus, D. L., "An Implicit Upwind Algorithm for Computing Turbulent Flows on Unstructured Grids," *Computers and Fluids*, Vol. 23, No. 1, 1994, pp. 1–22.
11. Theodorsen, T., "General Theory of Aerodynamic Instability and the Mechanism of Flutter," NACA Report Number 496, National Advisory Committee for Aeronautics, May 1934.
12. Giesing, J. P., Kalman, T., and Rodden, W. P., "Subsonic Unsteady Aerodynamics for General Configurations; Part II, Volume II – Computer Program N5KA," Air Force Flight Dynamics Laboratory Report No. AFFDL-TR-71-5, Part II, Vol. II, 1972.
13. Denegri, C. M., "Limit Cycle Oscillation Flight Test Results of a Fighter with External Stores," *Journal of Aircraft*, Vol. 37, No. 5, September-October 2000, pp. 761-769.
14. Schuster, D. M., and Byrd, J. E., "Transonic Unsteady Aerodynamics of the F/A-18E Under Conditions Promoting Abrupt Wing Stall," *Journal of Aircraft*, vol.41 no.3, May-June 2004, pp. 485-492.
15. Tijdeman, H., et al., "Transonic Wind Tunnel Tests on an Oscillating Wing with External Stores," AFFDL-TR-78-194, December, 1978.
16. Yates, E. C., Jr., "AGARD Standard Aeroelastic Configuration for Dynamic Response, Candidate Configuration I.-Wing 445.6," NASA TM 100492, August 1957.
17. Edwards, et al., "MAVRIC Flutter Model Transonic Limit cycle Oscillation Test," AIAA Paper AIAA-2001-1291, April 2001.